Static Analysis Bending Stress on Gear Tooth Profile By Variation of Gear Parameters With The Help of FEA

Mr. Kishor N. Naik

Student, M. E (Mechanical Cad/Cam),

Mechanical Engineering Department,

Jawaharlal Nehru Engineering College

Aurangabad, Maharashtra, India

Prof. Dhananjay Dolas

Associate Professor In Mechanical Engineering Department,

Jawaharlal Nehru Engineering College

Aurangabad, Maharashtra, India

Abstract -   Gears have wide variety of application. Their application varies from watches to very large mechanical units like the lifting devices and automotives. Gear generally fails when the working stress exceeds the maximum permissible stress.

The main objective of this paper is to analyze the bending stresses occur on the gear tooth profile of gear used in gear box of special purpose machine also effect on bending stress by variation of the gear parameters.
Face width and root radius are taken gear parameters, how stress redistribution are taken place by varying this parameter studied. The stresses are calculated with the help of the FEA this result are compared with the stresses calculated by Lewis equation.

For this work parametric modeling is done using Pro-e WF 5.0 and for analysis ANSYS 12.0 workbench is used. This work helpful to conclude effect of bending stress on gear tooth profile by variation of gear parameters also give the comparison of FEM method with analytical calculation.

Keywords: SPM (Special Purpose Machine) , FEA (Finite Element Analysis), IGES, Pro/E, Ansys 12.0

1.0 INTRODUCTION

Gears are the most common means of transmitting power in the modern mechanical engineering world. Spur gear is simplest and most widely used gear .The failure of gear is occurred due to exceed of bending stress in gear tooth profile. In this paper effect of bending stress on spur gear tooth profile by variation of gear parameter such as gear width and root radius are studied. With the help of FEA this result then compared with the Lewis equation.

 There are several kinds of stresses present in loaded and rotating gear teeth. We have to consider all the possibilities, so that the gears all the stresses with in the design limit. Generally stresses calculated in the gear design formula are not necessary true stress, can make it difficult to get correct answer on gear-tooth stresses, because it may not be know whether loaded uniformly distributed across the width and whether properly shared by the two or more pairs of the teeth that are in mesh at same time. So we have to make

right assumption that will allow for thing like stress concentration, residual stress, misalignment and tooth error, this means that the calculated stress is properly a not true stress. Each gear tooth may be considering as a cantilever beam, when it transmit the load, it subjected to loading [2]

Designing highly loaded spur gear for power transmission system that are both strong and quite requires analysis method that can easily be implemented and also provide information on contact and bending stresses. The finite element method is capable of providing this information, but the time required this model is large.

In order to reduce the modeling time, a preprocessor method that creates the geometry needed for finite element analysis may be used such that provided by Pro/Engineer. using API toolkit of Pro/Engineer one can generate model of three dimensional gear easily. In Pro/E, the geometry is saved as a file and then it can transferred from Pro/E to Ansys [1]

This literature contain the paper related to the stress analysis of the gear with the help of the FEM method some paper also related to failure mode of the gear. This literature provide me scope to analyze in static condition effect of bending stress on gear tooth profile with variation of gear parameter.

S. Kumar, K. Mishra, J. madan [1] they used the FEM method to analize the bending stress on gear tooth profile they observe that the contact and bending stresses major source of wear and tooth breakage respectively and subsequent failure of a gear. Shinde S.P,Nikam A.A, Mulla.T.S [3], in there paper bending stress analysis performed and trying to design the spur gear which resist the bending failure of the teeth. The bending stress calculated with help of the ANSYS. Kailash C. Bhosale [4] He give the comparison study between the Lewis equation and Ansys workbench. Sumit Agrwal [5] he used ANSYS as the FEA software and 3-Dimensional model of gear geometry. He found out bending stress for symmetric as well as asymmetric gear tooth.Samroeng Nephu [6] investigate the premature failure of a helical gear in reducer gear box for this he used the standard

investigative procedure of anlysis. Ivana Atanasovska, Vera Nikolic, D Dimitrijevic, D Momcilovic [7] describes the development of the finite element model for simultaneously monitoring the deformation and stress state of teeth flanks, teeth fillet and parts of the helical gear during the tooth pair meshing period. Ananda Kumar. Eriki, Mohd E. Musaffa [8] analize the spur gear crack propagation of path using the finite element method. Pravin M. Kinge, B.R. Kharde, B. R. Borkar [9] done the stress analysis of the gear box used in sugar industry. They used the ANSYS for stress analysis to find out the cause of failure of the gear and improve the life of the gear they found the reason of failure the gear is wear of gear of tooth edge.

2. METHODOLOGY & PROCESS

In this work gear used in special purpose machine is consider. Regular failure of gear occur due to the bending stresses which occur on gear tooth profile during working condition. So we carried out the work to find out value of face width and root radius of gear which produces minimum bending stress during working condition and life of gear get increases.
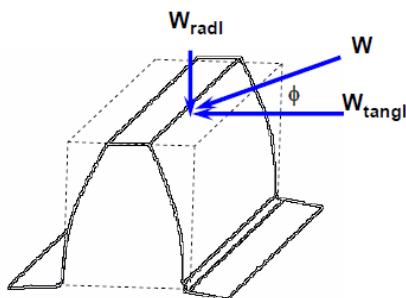
Gear having following specification.

Number of teeth(Z)	24
module (m)	4
Pitch Circle Diameter (D)	96
Base circle diameter	90
Pressure angle	20
Addendum circle diameter	104
Circular pitch	12.56
Thickness of tooth	8.1
Face width	10

Material of the gear steel having modulus of elasticity E=210000Mpa
Poisson ratio = 0.3. In this work gear box transmitting 4 KW power at 318 rpm,
So we can calculate Tangential load W_T

$$\text{Power in (KW)} = \frac{2 \, X \, \pi \, X \, N \, X \, T}{60 \, X \, 10^6},$$

$$\text{Torque (T)} = \frac{P(\text{kw}) X 60 \, X \, 10^6}{2 \, X \, \pi \, X \, N} \quad \text{........(1)}$$

$$\text{Tangential load } (W_T) = \frac{T}{\frac{D}{2}} \quad \text{...... (2)},$$

$$\text{Torque (T)} = \frac{4 X 60 \, X \, 10^6}{2 \, X \, \pi \, X \, 318} = 120177.8 \text{ Nm.....From eq (1)}$$

$$\text{Tangential load } (W_T) = \frac{120177.8}{\frac{96}{2}} = 2503.70 \text{ N.....From eq (2)}$$

2.1 GEOMETRIC MODELING

In this work parametric geometric modeling of spur gear is done by using the software Pro-e WF5.Parmetric modeling is nothing but modeling of each feature in such way that if we change dimension value of any one feature then according to that dimension of other get changed as per define relations.

First, open a new part file and create the basic gear parameters these are module, no of the teeth, pressure angle using Tools → Parameters. These basic parameters determine the all other parameters that define gear tooth profile. The other gear parameters are defined in terms of relationship of basic parameters using Tool → relation. By selecting the appropriate datum and commands like extrude, cut etc .The 3D geometric model of gear are created.
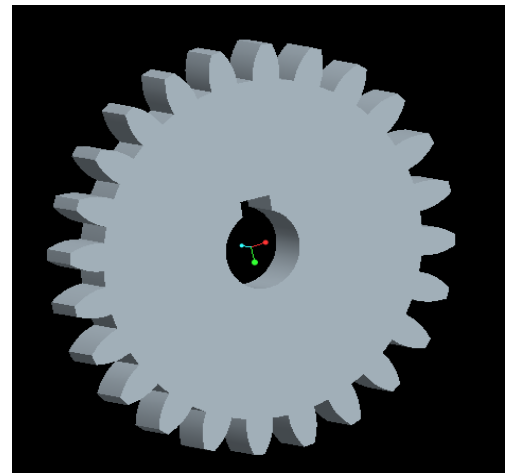

Fig 1. 3-D modeling in Pro/E WF5

After creating the 3D geometric model of involutes spur gear saved this model in IGES file so that it can be import to Ansys work bench for analysis
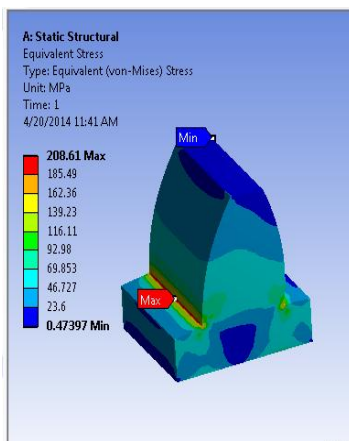
2.1 Finite Element Analysis

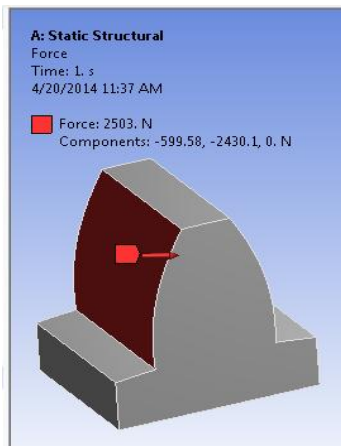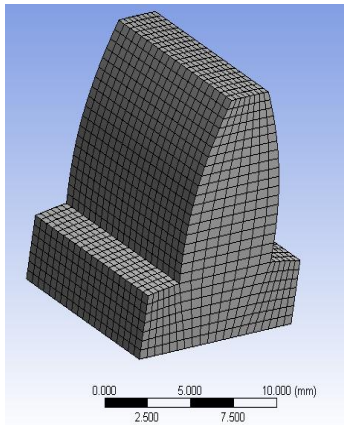For finite element analysis only tooth section is consider for analysis. The total analysis is done on the Ansys 12.0 workbench. A point load 2503N is applied at tip of the gear.
The basic procedure steps as follow.
1. Import the geometric model in the form of IGES.
2. Define the properties of material
3. Mesh the model.
4. Apply the boundary constrained and point load on model.
5. Solve for the vonmises (Equivalent) stress.

3. RESULT & DISCUSSION

Tabel No 2 shows the Ansys result of equivalent (vonmises stresses) for different face width and root radius.

As we taken root radius and the face width as the variable gear parametars the same procedure is done by changing the face width and the root radius we get the following result,

Sr No	Face width	Root radius	Equivalent stress
1	10	0	208.61
2	10	1	232.55
3	10	2	168.63
4	10	3	153.81
5	10	4	152.51
6	15	0	128.58
7	15	1	156.27
8	15	2	113.82
9	15	3	95.24
10	15	4	101.15
11	20	0	88.94
12	20	1	112.07
13	20	2	82.53
14	20	3	74.90
15	20	4	73.43
16	25	0	68.26
17	25	1	89.65
18	25	2	65.60
19	25	3	58.74
20	25	4	57.45

Tabel No 2.Ansys result for different face width & root radius
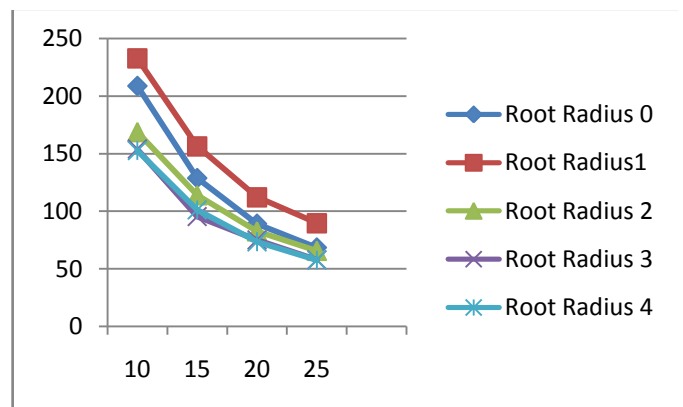
Chart No1. Bending stress vs Face width.

Fig 2 shows the Ansys FEA result of gear tooth having root radius 4 and different face width.

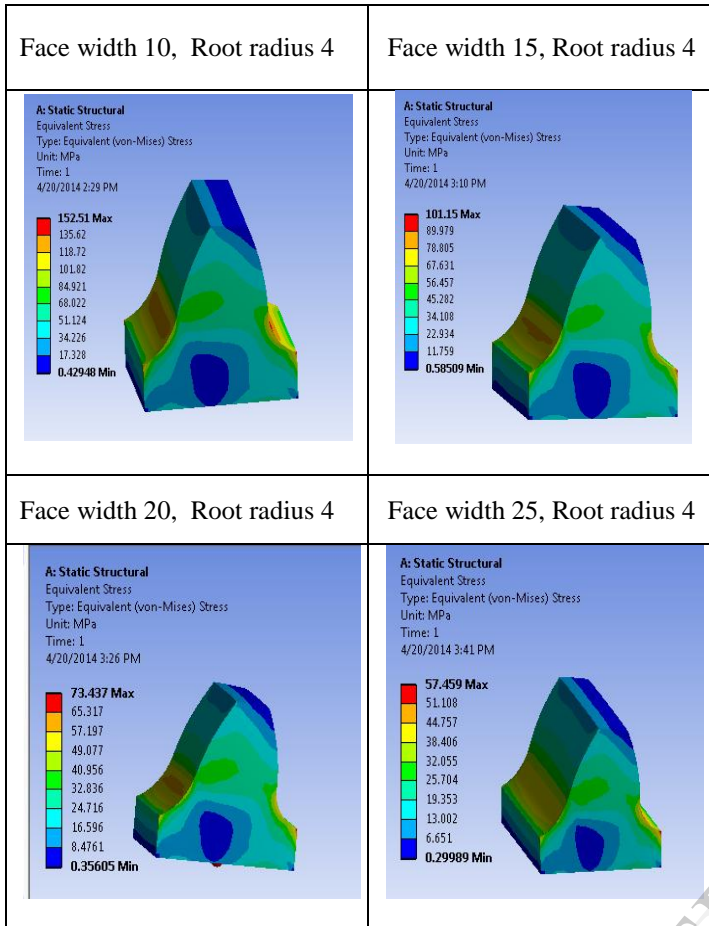| Face width 10, Root radius 4	Face width 15, Root radius 4
A: Static Structural Equivalent Stress Type: Equivalent (von-Mises) Stress Unit: MPa Time: 1 4/20/2014 2:29 PM 152.51 Max 135.62 118.72 101.82 84.921 68.022 51.124 34.226 17.328 0.42948 Min	A: Static Structural Equivalent Stress Type: Equivalent (von-Mises) Stress Unit: MPa Time: 1 4/20/2014 3:10 PM 101.15 Max 89.979 78.805 67.631 56.457 45.282 34.108 22.934 11.759 0.58509 Min
Face width 20, Root radius 4	Face width 25, Root radius 4
A: Static Structural Equivalent Stress Type: Equivalent (von-Mises) Stress Unit: MPa Time: 1 4/20/2014 3:26 PM 73.437 Max 65.317 57.197 49.077 40.956 32.836 24.716 16.596 8.4761 0.35605 Min	A: Static Structural Equivalent Stress Type: Equivalent (von-Mises) Stress Unit: MPa Time: 1 4/20/2014 3:41 PM 57.459 Max 51.108 44.757 38.406 32.055 25.704 19.353 13.002 6.651 0.29989 Min

Fig 2.0 Ansys result.

According to Chart No 1 we conclude that gear having face width 25, and root radius 4 produces lower bending stress so that we cross check this result with theoretical calculation by using Lewis equation result.

3.1 Comparison of Result

To check Ansys results we cross check this result with analytical method. For analytical method we use Lewis equation .This is the classic method of estimating the bending stresses in a gear tooth. It models a gear tooth taking the full load at its tip as a simple cantilever beam
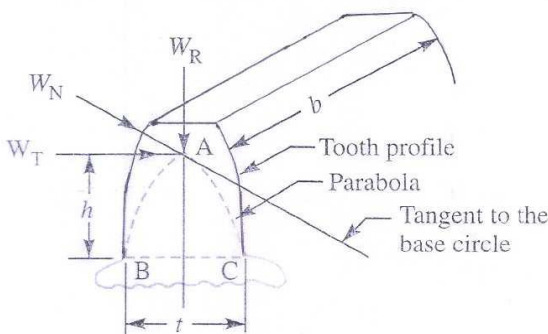

Fig 3 Load acting on gear tooth profile

Assumption of the Lewis Equation.
1. Each tooth may be considered as cantilever beam.
2. The force W_T between the tooth surfaces is normal to the surface acting at the tip of tooth.
3. The effect of radial component W_R radial neglected.
4. The effect of stress concentration neglected.
5. At any time only one pair of teeth is in contacts and takes the load.

M = is the maximum bending moment at critical section BC.
W_T = Tangential load acting at the tooth profile.
h = Length of the teeth.
Y = Half of the thickness of the tooth at critical area BC.
I = Moment of the inertia about center of line of the tooth.
b = Width of gear face.

$$\sigma_w = M \times \frac{Y}{I} \quad \ldots\ldots\ldots (3)$$

$$M = W_T \times h \quad \ldots\ldots\ldots\ldots (4)$$

$$Y = \frac{t}{2} \quad \ldots\ldots\ldots\ldots (5)$$

$$I = \frac{bt^3}{12} \quad \ldots\ldots\ldots\ldots (6)$$

$$\sigma_w = \frac{\frac{(W_T \times h) \times t}{2}}{\frac{bt^3}{12}} \quad \ldots\ldots\ldots (7)$$

$$W_T = \sigma_w \times b \times \frac{t^2}{h \times 6}$$

$$\sigma_w = \frac{(W_T \times h) \times 6}{bt^2}$$

Calculation

M = 2503.70X 6.4 =16023.71 Nm …From eq (4) and eq (2)

$Y = \frac{8.1}{2} = 4.05$ ……..From eq (5)

$I = \frac{10 \times 8.1^3}{12} = 442.86$ ……From eq (6)

$\sigma_w = M \times \frac{Y}{I} = 16023.71 \times \frac{4.05}{442.86} = 146.53$ N/m.

By doing the same procedure we get the bending stresses for different face width and root radius 4. As shown in table 3.

Sr No	Face Width	ANSYS Calculated Equivalent stresses	Theoretical Lewis Equation Calculated stresses	% Of Correctness
1	10	152.51	146.53	95.91
2	15	101.15	97.69	96.54
3	20	73.43	73.26	100.17
4	25	57.45	58.61	98.84

Tabel No 2. Result comparison

CONCLUSIONS

From chart we can say that gear having the face width 25 and root radius 4 produces minimum bending stress. Bending stresses at contact region decreases with increase the face width. Bending stresses at contact region decreases with increase the root radius.FEA results are found to be close agreement of the Lewis equation.

REFERENCES

1. Sunil Kumar, K.K. Mishra, Jatinder Madan, Stress Analysis Of Spur Gear Using FEM method , National Conference of Advancements and Futuristic Trends in Mechanical and Material Engineering (Feb 19.20.2010)
2. Sushil Kumar Tiwari, Upendra Kumar Joshi, Stress Analysis of moving involute spur gear teeth ISSN:2278-0181
3. Shinde S.P, Nikam A.A, Mulla T.S, Static Analysis of spur gear using finite element analysis, ISSN:2278-1784,PP:26-31
4. Kailash C. Bhosale, Analysis of bending strength of the helical gear by FEM, ISSN 2222-1727(Paper) ISSN 2222-2871(Online) Vol 2, No 4, 2011
5. Sumit Agrwal, R.L. Himte, Evalution of Bending Stress at Fillet Region of an Asymmetric Gear with a Hole as a stress relieving feature using a FEA Software ANSYS. International Journal Of Computer Application Volume 51-No 8, Aug 2012
6. Samroeng Netpu, Panya Scrichandr, Failure Analysis Of Halical Gear, The First TSME International Conference on Mechanical Engineering 20-22 Oct 2010,Ubon Ratchathani
7. Ivana Atanasovska, Vera Nikolic, Dejan Dimitrijevic, Dejan Momcilovic, Finite Element Model for Stress Analysis and Nonlinear Contact Analysis of Helical Gears, Scientic Technical Review, Vol LVIX,No-1,2009
8. Ananda Kumar. Eriki, Mohd Edilan. Musaffa, Spur Gear Crack Propagation Path Analysis Using Finite Element Method
9. Pravin M. Kinge, B.R. Kharde, B. R. Borkar, Stress Analysis Of Spur Gearbox, ISSN2250-3498,Vol.2, No 3, June 2012.
10. Bharat Gupta, Abhishek Choubey, Gautam V. Varde , Contact Strss Analysis Of Spur Gear, ISSN:2278-0181,Vol.1 Issue 4, June-2012.